LATERAL LOAD RESISTING BEHAVIOUR OF EXISTING RAILWAY BRIDGE PIERS

A THESIS

Submitted by

KALE MAHESH BABU

for the award of the degree

of

MASTER OF TECHNOLOGY

DEPARTMENT OF CIVIL ENGINEERING
NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA 769008

May 2015

THESIS CERTIFICATE

This is to certify that the thesis entitled "**LATERAL LOAD RESISTING BEHAVIOUR OF EXISTING RAILWAY BRIDGE PIERS"** submitted by **Kale Mahesh Babu** to the National Institute of Technology, Rourkela for the award of the degree of Master of Technology is a bonafide record of research work carried out by him under my supervision. The contents of this thesis, in full or in parts, have not been submitted to any other Institute or University for the award of any degree or diploma.

Project Guide

Rourkela-769 008
Date:

Dr. Pradip Sarkar
Associate Professor
Department of Civil Engineering

ACKNOWLEDGEMENTS

I am deeply indebted to **Dr. Pradip Sarkar,** Associate Professor, my guide, for the motivation, guidance, tutelage and patience throughout the research work. I appreciate his broad range of expertise and attention to detail, as well as the constant encouragement he has given me. There is no need to mention that a big part of this thesis is the result of joint work with him, without which the completion of the work would have been impossible.

My sincere thanks to the Head of the Civil Engineering Department **Prof. S. K. Sahu**, National Institute of Technology Rourkela, for his advice and providing necessary facility for my work. I am also grateful to **Prof. Robin Davis P**, for his valuable suggestions and timely cooperation during the project work. I am very thankful to all the faculty members and staffs of Civil engineering department who assisted me in my project work, as well as in my post graduate studies.

I would like to take this opportunity to thank all the people who directly or indirectly helped me to complete the project work.

Kale Mahesh Babu
Roll No: 213CE2058
M. Tech. (Structural Engineering)

ABSTRACT

Most of the sub-structures of new railway river bridges in the state of Odisha are built with solid mass concrete gravity piers and abutments. These piers do not have steel reinforcement to bear the load as it does not subject to any tensile stress under regular type of loading. Safety of these piers is of major concern during high magnitude earthquake as frequent occurrence of such earthquakes is observed in India in recent times. Failure of pier may result in loss of functionality of Railway Bridge leading to the cut down of rail communication line for an indefinite amount of time and a huge loss to the society.

This study aims to assess the vulnerability of the solid gravity bridge piers which forms the important component of railway bridges as the load transfer between substructure and superstructure takes through them. In the present study seven existing piers from the state of Odisha are evaluated using free vibration analysis and nonlinear static (pushover) analysis.

Free-vibration analysis of the bridge pier shows that the mass participation of fundamental mode is always below 50%. Also, the cumulative mass participation for first six mode is found to be less than 80% for all the selected bridge pier. This indicates the significant contribution of higher modes. The pushover analysis indicates the brittle mode of failure of all the bridge piers at ultimate load. This is due to poor energy dissipation capacity of the mass concrete used for building these structures.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

NOTATIONS

L	Length of the rectangular portion of the pier
B	Width of the rectangular portion of the pier
D	Diameter of the semicircular portion of the pier
h	Height of the pier
G	potential plastic flow
σ_{to}	Uniaxial tensile stress at failure
$\overline{p}$	Hydrostatic pressure stress
$\bar{q}$	Mises equivalent effective stress
ψ	dilatency angle measured in the *p*-*q* plane at high confining pressure
ϵ	Eccentricity of the plastic potential surface
σ_{cu}	Ultimate compressive strength
$\tilde{\varepsilon}_c^{in}$	Inelastic strain in compression
ε_c	Total strain in compression
ε_{oc}^{el}	Elastic strain in compression
σ_{cu}	Ultimate compressive strength
$\tilde{\varepsilon}_t^{ck}$	Cracking strain in Tension
ε_t	Total strain in tension
ε_{ot}^{el}	Elastic strain in Tension
d_c	Compression Damage parameter
d_t	Tension Damage Parameter
UX	Cumulative mass participation in Translational X- direction
UY	Cumulative mass participation in Translational Y- direction
W	Weight of the pier

V_B Base Shear

δ Top displacement of the pier

CHAPTER 1
INTRODUCTION

1.1 BACKGROUND AND MOTIVATION

Indian Railway has spent huge amount of money in last five years for doubling the railway lines in order to enhance capacity and generate returns. A number of new rail bridges has come up due to these project. Superstructure of most of these steel bridges is built using railway standard design. However, the sub-structure of these bridges are designed individually considering the site conditions. A survey on these new bridges in the state of Odisha revealed that most of the sub-structures of these bridges are built with solid mass concrete gravity piers and abutments supported by either pile foundations or well foundations. Fig. 1.1 shows a typical mass concrete solid gravity bridge pier constructed by Indian railways. These mass concrete bridge piers have certain amount of skin reinforcement to resist the stresses developing on the surface of the pier due to shrinkage. However, these piers do not have steel reinforcement to bear the load as it does not subject to any tensile stress under regular type of loading.

Fig. 1.1: Typical mass concrete solid gravity bridge pier of Indian Railways

A frequent occurrence of natural hazards (such as earthquake, cyclone, etc.) of high magnitude in Indian subcontinent raised the question of safety of these piers under such natural hazards. Failure of pier results in loss of functionality of Railway Bridge leading to the cut down of rail communication line for indefinite amount of time and huge loss to the society. Therefore it is a very important task to evaluate the safety such bridge piers against different natural hazards. This is the primary motivation of the present Study. As a first step towards this, the solid mass concrete railway bridge piers are studied for their behavior against seismic loading.

1.2 REVIEW OF LITERATURE

A number of study on the seismic evaluation of road RC bridge pier (Spyrakos, 1992; Lee et al., 2005; Kim et al., 2006; Wang et al., 2008; Faria et al., 2000;, Shinge, 2010); Spyrak, 1992; Shim et al., 2008; and pre-stressed concrete bridge pier (Wang et al., 2011) are reported in literature.

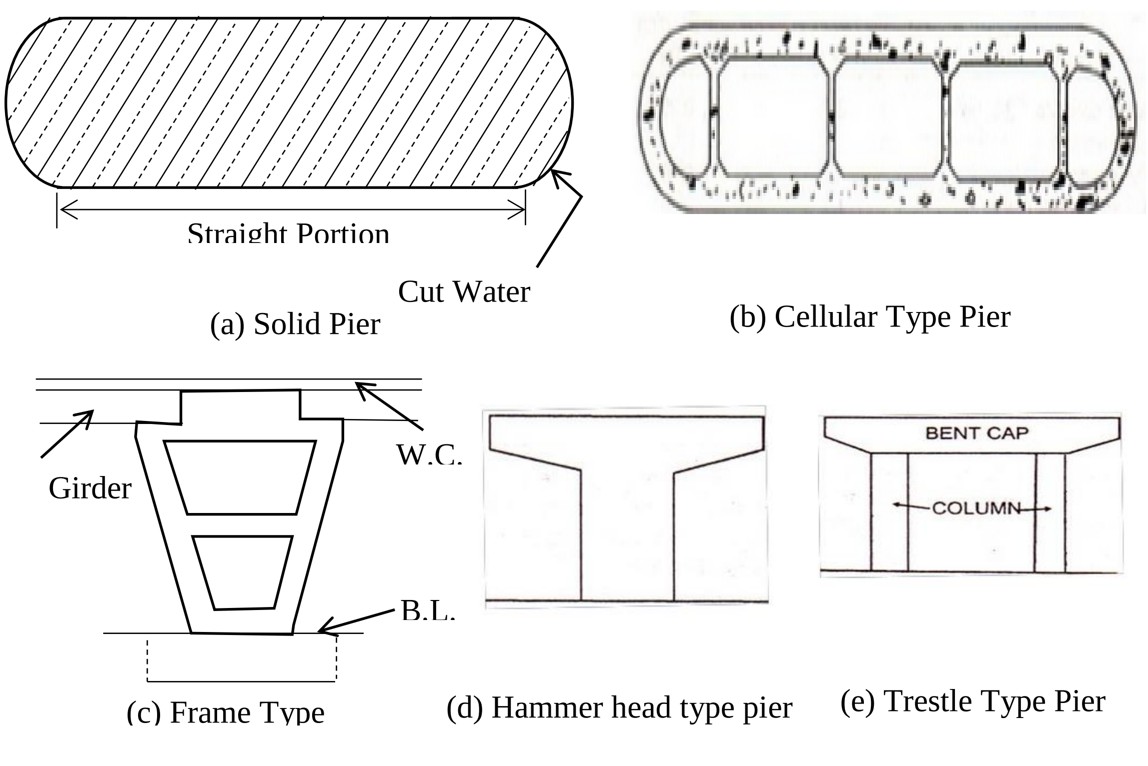


Fig 1.2: Different types of piers

These studies are based on both linear and nonlinear analysis of piers subjected to equivalent static or dynamic seismic loading. Most of these studies have considered the whole bridge (consisting a number of piers with bridge deck) for analysis. However, there are few studies that have arrived at conclusion based on analysis of individual bridge piers. The analysis of single individual bridge pier is reported to have conservative estimation of forces. Railway bridge piers which are an important component of railway line has been paid very less research attention. There are different types of bridge piers used in Indian Railways: (a) Solid piers, (b) Cellular type pier, (c) Framed type bridge pier, (d) Hammer-head pier, and (e) Trestle type pier. The first two among them are made of mass concrete whereas the remaining three are usually made of reinforced concrete. Fig. 1.2 presents different type bridge piers used in the Indian Railways. An extensive literature review on the seismic evaluation of bridge pier revealed no published research work on the solid mass concrete gravity bridge pier.

1.3 OBJECTIVES

Based on the literature review presented above the objectives of the present study is identified as follows:

(i) To study the free vibration analysis of solid mass concrete gravity bridge pier

(ii) To study the nonlinear force deformation behavior of solid mass concrete gravity bridge pier

1.4 METHODOLOGY

a) A thorough literature review to understand the seismic evaluation of building structures and application of push over analysis.

b) Select an existing railway bridge pier with geometrical and structural details.

c) Model the selected railway bridge pier in computer software ABAQUS.

d) Carry out free vibration analysis and nonlinear static (pushover) analysis of the model and arrive at a conclusion.

1.5 SCOPE OF THE STUDY

1) The selected railway bridge piers are modelled with M25 concrete.
2) The bottom end of the pier is assumed to be fixed at the top of the foundation.
3) Loading is considered only in one direction.
4) Failure at other locations like joints and connection of structural members are not considered.
5) The present study is based on static non-linear pushover analysis

1.6 ORGANISATION

This introductory chapter presents the background, review of literature, objectives, scopes and methodology of the present study.

Chapter 2 discusses details about free vibration analysis and pushover analysis procedures.

Chapter 3 presents the geometry of the selected railway bridge piers and discusses the issues related to structural and material modelling in ABAQUS.

Chapter 4 presents the results obtained from free-vibration analysis and pushover analysis. It also presents the discussions and interpretations of the results.

Finally, in Chapter 5 the summary and conclusions are given.

CHAPTER 2

ANALYSIS METHODS

2.1 GENERAL

Free vibration response of structures is very important to assess their behaviour when subjected to dynamic loading like earthquake and wind. Also, when the structures are subjected to strong earthquake or wind storm, they exhibit inelastic behaviour, which cannot be assessed by elastic analysis. A non-linear analysis evaluates the performance of the structures taking into account the post-elastic behaviour and predicts the vulnerability of the structure. The present study conducts free vibration analyses and pushover analysis using finite element software ABAQUS. This chapter describes the detailed procedures of free-vibration analysis and pushover analysis used in the present study. Finally, this chapter presents brief discussions on finite element analysis.

2.2 FREE-VIBRATION ANALYSIS

The equations of motion associated with the free vibration response of a structure are given by:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{0}$$

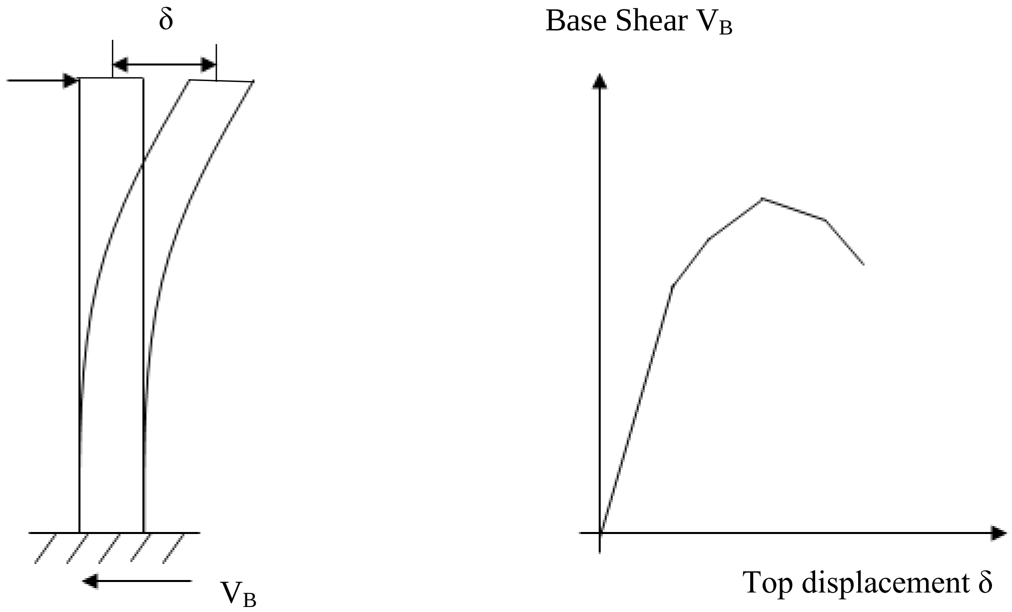
Here, **M** is the diagonal mass matrix, **K** is the stiffness matrix, $\ddot{\mathbf{u}}$ and **u** are the acceleration and displacement vectors, respectively.

The objective of free-vibration analysis is to obtain the natural frequencies, mode shapes and other modal parameters.

2.3 PUSHOVER ANALYSIS

The traditional approach employs the linear static analysis, in which an elastic analysis is used to determine the lateral seismic forces, then by use of certain response reduction factors these are converted to inelastic forces. Though the approach had several shortcomings, it was accepted for its simplicity, and the lack of alternative practical approaches (Imbsen et al., 1996).

Displacement based methods are the widely used methods of analysis for predicting the demand imposed by earthquakes on the structures. Evaluation of seismic performance is given importance in these methods. The main purpose of this analysis is to develop a pushover curve of the structure which reasonably estimates inelastic behaviour. Fig 2.1 represents the schematic representation of pushover analysis.



Structure Model

Pushover Curve

Fig 2.1: Schematic representation of pushover analysis

2.4 FINITE ELEMENT ANALYSIS

FEA is a powerful numerical technique to find an approximate solutions to a wide range of "real-world" complex engineering problems having general boundary conditions. FEA has become an essential step in the design or modelling of a physical phenomenon in various engineering disciplines. Fig 2.2 shows the flow chart of development of computational model in FEM analysis.

The general procedure to solve a problem in FEM is

1) Model the structure with suitable geometry and material properties.
2) Discretize the model into elements by suitably selecting the type of element.
3) Apply the boundary conditions and force vectors.
4) Stiffness matrix for the element will be developed by the software and combined to form global stiffness matrix and force and displacement matrix will be developed.
5) Finally solutions to the problem are obtained.

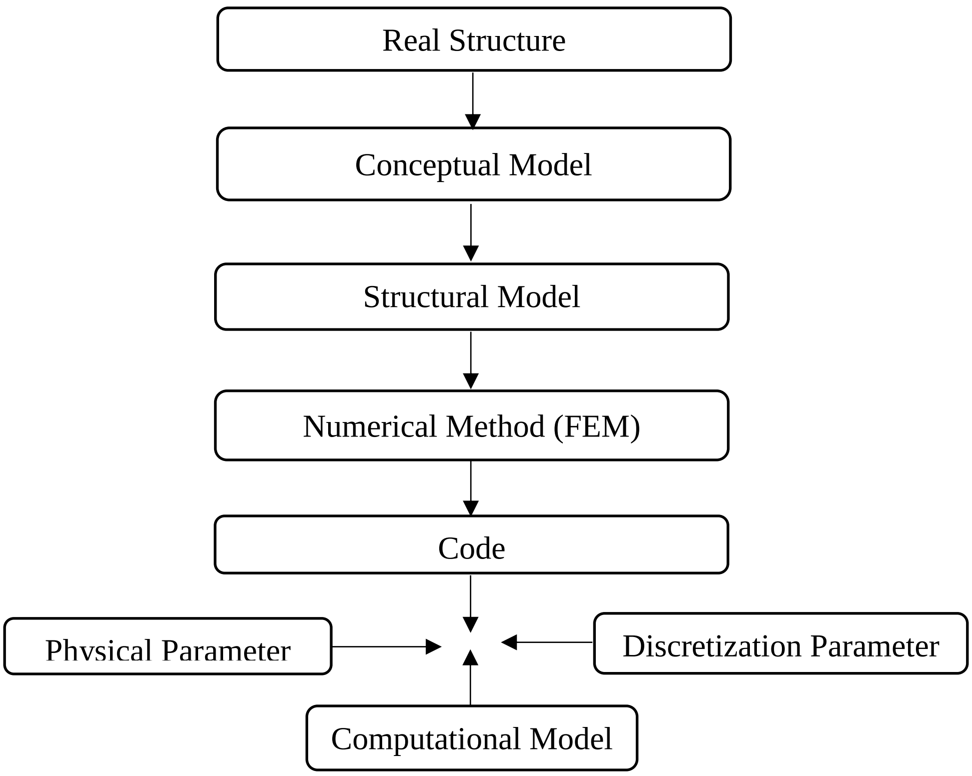


Fig 2.2 Flow Chart showing development of computational model in FEM (ref: Onate 2009**)**

2.5 ANALYSIS STEPS INVOLVED IN ABAQUS

In Abaqus modelling and analysis includes following three steps:

1. Pre-processing
2. Simulation
3. Post processing

2.5.1 Pre-processing

In this step model of the physical problem is defined and an abaqus input file (job.inp) in generated. Basic key points like material properties, boundary condition, load, contact, mesh are defined here.

2.5.2 Simulation

The simulation is normally run as a background process. In this step already generated abaqus input file solves the numerical problem defined in the model. For example, output from a stress analysis problem includes displacement and stress values which stored in binary files in simulation which are further to be used in post-processing. The output file is generated as job.odb.

There are three phases in simulation stage

- Analysis step
- Load increment
- Iteration

In first phase, we have to define steps which generally consists of loading option, output request. Output request describes the values of required parameters like displacement, stress, strain, reaction force, bending moment etc.

Second phase is the increment step, in which first load increment is to be defined by user and the subsequent increments will be chosen by abaqus automatically. After each load increment the structure is in equilibrium and corresponding output request values are to be written to the output database file.

In iteration step, approximate equilibrium solution in each increment is found out. If the structure is not in equilibrium after iteration, abaqus tries further iteration till closest possible equilibrium is obtained or the residual force is less than the given tolerance value.

2.5.3 Post processing

Once the simulation is done and the fundamental variables like stress, displacement, reaction forces are calculated, the results can be evaluated using Visualization module of abaqus. The visualization module has variety of options to display the results such as animation, colour contour plots, deformed shape plots and *X-Y* plots.

2.6 SUMMARY

This Chapter presents the analysis methods used in this study i.e., free vibration analysis, pushover analysis and finite element analysis. This chapter also presents about the analysis steps involved in abaqus software.

CHAPTER 3

STRUCTURAL MODELLING

3.1 INTRODUCTION

The study in this thesis is based on nonlinear analysis of selected railway bridge piers. This chapter presents a summary of various parameters defining the computational models. In structural members, several degradation types takes place like crushing, cracking, damage, spalling of concrete and yielding of reinforcement. Hence, to accurately model the non-linear properties of the materials and structural components plays a vital role in non-linear analysis. In the present study, piers were modelled with inelastic concrete damaged plasticity model (CDPM) in abaqus.

The response of structure under loading is critical in estimating its efficiency and safety. Experimental analysis is widely used since it gives real time response but it is time consuming and costly. The use of finite element packages makes the analyses cost-effective and we can understand the response of the structure. For the purpose of the modelling it is important to understand the element types their features and behavior. Therefore it is necessary to have an idea about the different types of elements used for modelling in finite element software.

3.2 STRUCTURAL DETAILS

A survey was conducted to obtain the structural details of existing solid mass concrete gravity bridge pier. As a result details of such bridge piers from seven newly constructed railway river bridges were obtained. These bridge piers have rectangular section with semi-circular ends and slanted in both directions. Fig. 3.1 presents the typical elevations along flow and traffic directions and the typical plan of these bridge piers. The side slope of these bridge piers is found to be equal in both the directions. All of these available piers are rigidly connected to the pile cap using a number of dowel bars. The pile caps, in all the cases, are supported by multiple piles. All the railway bridges considered in this study are multi-span bridges and therefore has multiple piers. The piers of one bridge are identical in shapes and sizes except for the length. Length of the piers is varying according to the river profile. One representative pier from each of the seven bridges are considered for analysis. Table 3.1 presents the dimensions of selected seven piers. Mass concrete of M-25 grade was used to build these piers.

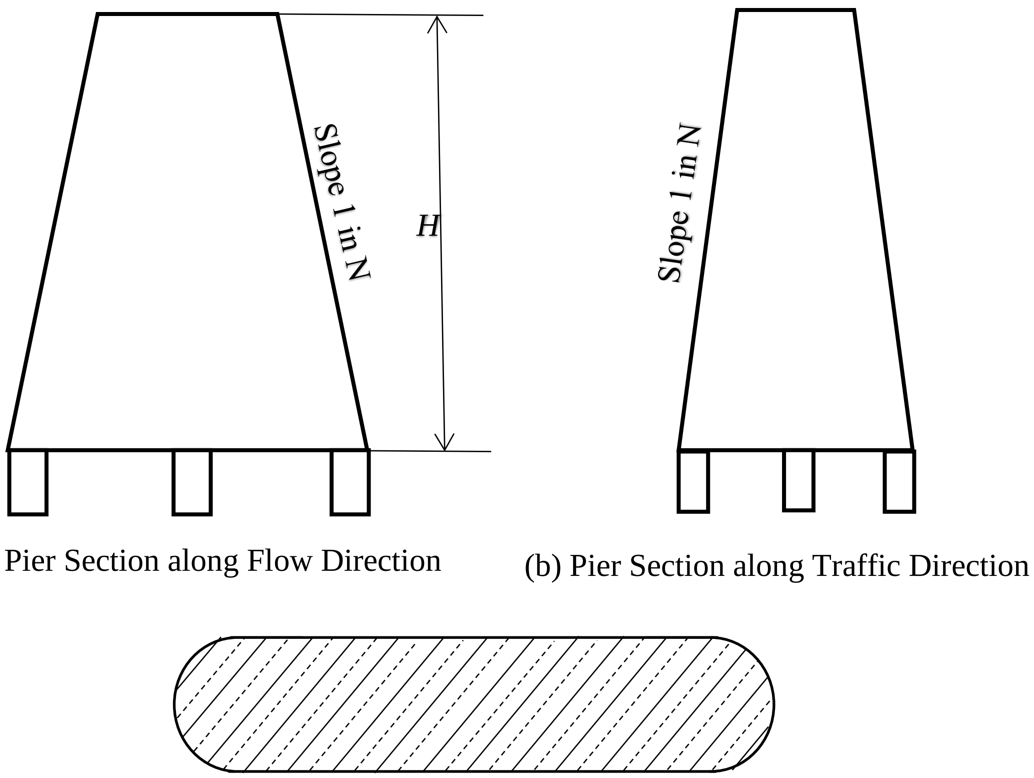


Fig. 3.1: Plan and Elevations of typical Solid Gravity Pier used in this study

Table 3.1: Details of the selected piers

SL. No.	Height, H (m)	Base Dimension (m)		Top dimensions (m)		Slope (in both dirn.)
		Rectangular Portion (L×B)	Diameter of semi-circular portion, (D)	Rectangular Portion (L×B)	Diameter of Semi-circular portion, (D)	
1	7.500	4×2.50	2.50	4×1.5	1.5	1 in 15
2	10.500	4×2.90	2.90	4×1.5	1.5	1 in 15
3	8.402	4×2.62	2.62	4×1.5	1.5	1 in 15
4	11.250	4×3.75	3.75	4×1.5	1.5	1 in 10
5	16.875	4×4.75	4.75	4×2.5	2.5	1 in 15
6	12.000	4×3.90	3.90	4×1.5	1.5	1 in 10
7	12.375	4×4.15	4.15	4×2.5	2.5	1 in 15

3.3 ELEMENT USED IN ABAQUS

Abaqus element library provides a large variety of elements in modelling different geometries and structures like beam elements, brick elements, truss elements, membrane elements, shell elements, quadrilateral elements. Fig 3.2 shows the elements available in the abaqus element library.

Beam elements:

2 node linear beam element in plane (designation B21 in ABAQUS): This type of elements has two nodes and each node has three degrees of freedom. Used for plane stress analysis.

2 node linear beam in space (designation B31): It is an element used for simple stress-strain analysis. It has two nodes and six degrees of freedom at each node at space.

3 node quadratic beam in plane-(designation B22): It is quadratic beam element used for stress analysis. Each element has three nodes. Each nodes carries three degree of freedom.

3 node quadratic beam in space-(designation B32): It is a quadratic element in space which has three nodes, six degrees of freedom at each node. It is used for stress concentration and for analysis of beam in space or frame in space.

The solid element library includes isoparametric elements: quadrilaterals in two dimensions and **“bricks” (hexahedra)** in three dimensions. These isoparametric elements are generally preferred for most cases because they are usually the more cost-effective of the elements that are provided in ABAQUS. They are offered with first- and second-order interpolation. Standard first-order elements are essentially constant strain elements: the isoparametric forms can provide more than constant strain response, but the higher order content of the solutions they give is generally not accurate and, thus, of little value. The second-order elements are capable of representing all possible linear strain fields. Therefore, it is generally recommended that the highest-order elements available be used for such cases: in ABAQUS this means second order elements.

8 node linear brick element reduced integration-(designation C3D8R): It is an 8 node 3D brick element and on the elemental nodes only the degrees of freedom are calculated and at other nodes the values are obtained by interpolating with the nodal values. Used for higher order beam analysis.

20 node quadratic brick element, reduced integration-(designation C3D20R): It is used for three dimensional model analysis. Degrees of freedom are calculated only element nodes. At any other point in the element, the values are obtained by interpolating from the nodal values. Number of element nodes determines the interpolation order. Elements with mid side nodes, such as the 20-node brick use quadratic interpolation and are often called quadratic elements

3 node quadratic 3-D truss element-(designation T3D3): This type of element has three nodes. It is a basically bar element. For modelling of reinforcement this type of element is used.

A 2-node linear 3-D truss-(T3D2): This type of element has two nodes. It is used as bar elements and for reinforcement modelling. It is required for truss member modelling.

8node quadrilateral membrane, reduced integration-(designation M3D8R): Membrane standard quadratic element. Thin surfaces in space are represented by these type of elements that offer strength in the plane of the element but have no bending stiffness.

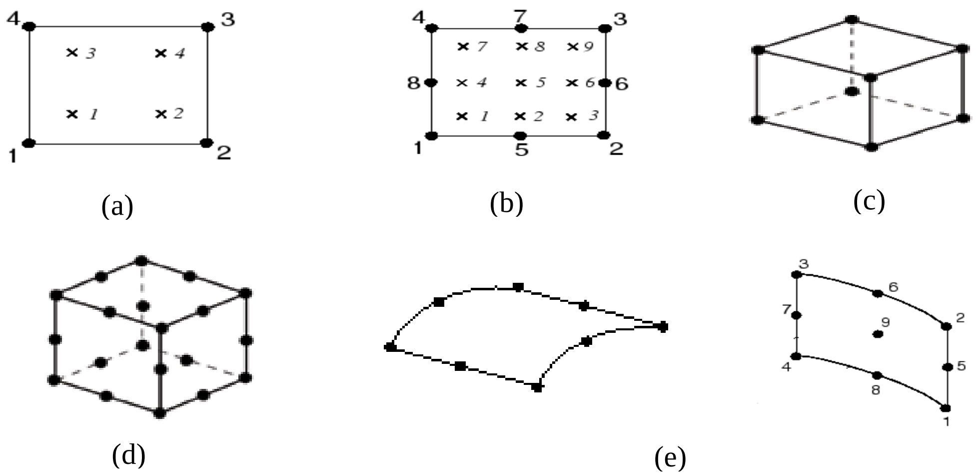


Fig 3.2: Finite elements-(a) linear element,; B21,B31 (b) quadratic element; B22,B32 (c) C3D8R (d) C3D20R (e) M3D8R

3.4 GEOMETRIC MODELLING

- First, in the part module, the base section of the pier is drawn and extruded to the height, H of the selected pier with slope 1 in N.
- In the property module, then material properties i.e., concrete damaged plasticity model parameters are given and assigned to the section..
- In the step module, step is created for the type of analysis. In the present study static

procedure is selected and geometric non-linearity is taken into account by selecting the Nlgeom-on.

Step ⟶ General ⟶ Static ⟶ Nlgeom

- In the load module, boundary conditions are created. Fig 3.3 shows the boundary conditions applied to the structure. The bottom end of the pier is assumed fixed and top is displaced to certain displacement. The displacement is applied in incremental manner.

In the mesh module module, the structure is discretized into finite number of elements. The structure is discretized using C3D8R element. It is 3D element with 8 nodes with six degrees of freedom with three translations at each node translations in the nodal x, y, z directions and rotational along nodal x, y, z directions. At other nodes the displacements, stress, strain values are calculated by interpolating with the nodal values.

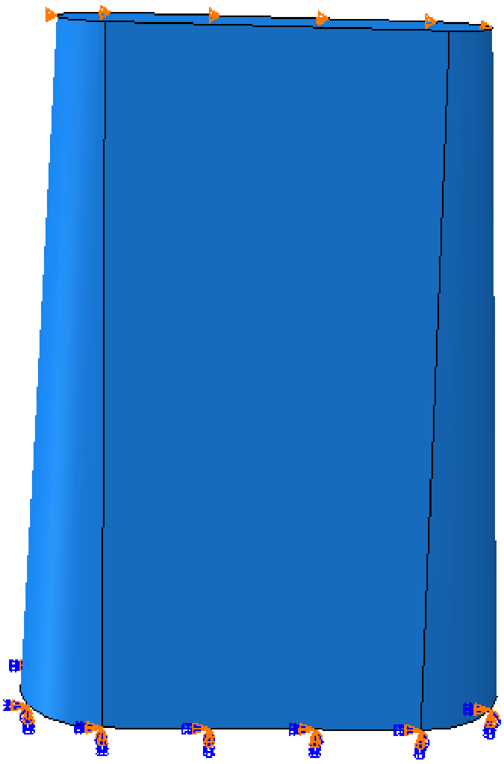


Fig 3.3: Pier model in abaqus

3.5 MATERIAL MODELLING

The exact behaviour of the structure cannot be estimated by the elastic material properties under high intensity loads. Hence non-linear material properties have to be determined for these type of types. In this study Concrete Damaged Plasticity Model is used for material modelling.

3.5.1 Concrete Damaged Plasticity Model (CDPM)

The two important failure mechanisms of concrete is assumed to be compression crushing and tensile cracking. This model represents the complex behaviour of the material by considering the isotropic damaged elasticity with isotropic tensile and compressive plasticity. Lubliner et al. 1989 first proposed this model and later was consolidated by Lee and Fenves 1998. Concrete Damaged Plasticity Model was put into implementation in FEM software (ABAQUS). The potential plastic flow is assumed in the model G and it is the Drunker-Prager hyperbolic function:

$$G = \sqrt{(\in \sigma_{to} \tan \psi)^2 + \bar{q}^2} - \bar{p} \tan \psi$$

σ_{to} is the uniaxial tensile stress at failure, $\bar{p}$ is the hydrostatic pressure stress, $\bar{q}$ is the Mises equivalent effective stress, ψ is the dilatency angle measured in the *p*-*q* plane at high confining pressure and ϵ is an eccentricity of the plastic potential surface. This model makes use of a yield state centred on loading function F proposed by Lubliner et al. 1989 with the changes made by Lee and Fenves 1998 to account for evolution of strength under tension and compression in the form:

$$F = \frac{1}{1-\alpha} (\bar{q} - 3\alpha \bar{P} + \beta(\tilde{\varepsilon}^{pl} \langle \sigma_{max} \rangle - \gamma \langle -\sigma_{max} \rangle) - \bar{\sigma}_c \tilde{\varepsilon}^{pl} = 0$$

$$\alpha = \frac{(\sigma_{bo}/\sigma_{co}) - 1}{2(\sigma_{bo}/\sigma_{co} - 1)} \ ; 0 < \alpha < 0.5$$

$$\beta = \frac{\bar{\sigma}_c(\tilde{\varepsilon}_c^{pl})}{\bar{\sigma}_t(\tilde{\varepsilon}_t^{pl})} (1-\alpha) - (1+\alpha)$$

$$\gamma = \frac{3(1-K_c)}{2K_c - 1}$$

Factor α depends on the ratio of the biaxial and uniaxial compressive strengths $(\sigma_{bo}/\sigma_{co})$, K_c is the ratio of magnitude of the deviatory stress in uniaxial tension to the uniaxial compression and it should satisfy the limit of 0.5$< K_c < 1.0$. Thus it clearly understood that the concrete behaviour depends on four constitutive parameter $(K_{c\,,}\psi, (\sigma_{bo}/\sigma_{co})\,, \epsilon)$. The other parameters include the stress-strain behaviour of concrete in compression and tension. Concrete exhibits softening behaviour and stiffness degradation that leads to convergence difficulties, hence to allow stresses to be outside the yield surface a viscosity factor μ has been added to the CPDM in abaqus.

The value of K_c is determined by Mohr Coulomb yield surface function and its value controls the shape of the yield surface if K_c = 1 the yield surface is a circle and K_c = 0.5 the yield surface is a triangle. (fig 3.4)

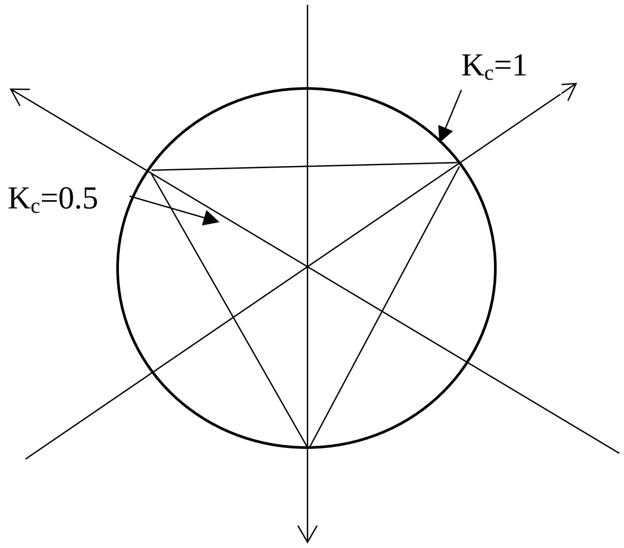


Fig 3.4: Yield Surface for K_c = 1 and K_c = 0.5

Dilatancy angle (ψ): It is the phenomenon of change of the inelastic volume to plastic deviation in a frictional material during shearing.

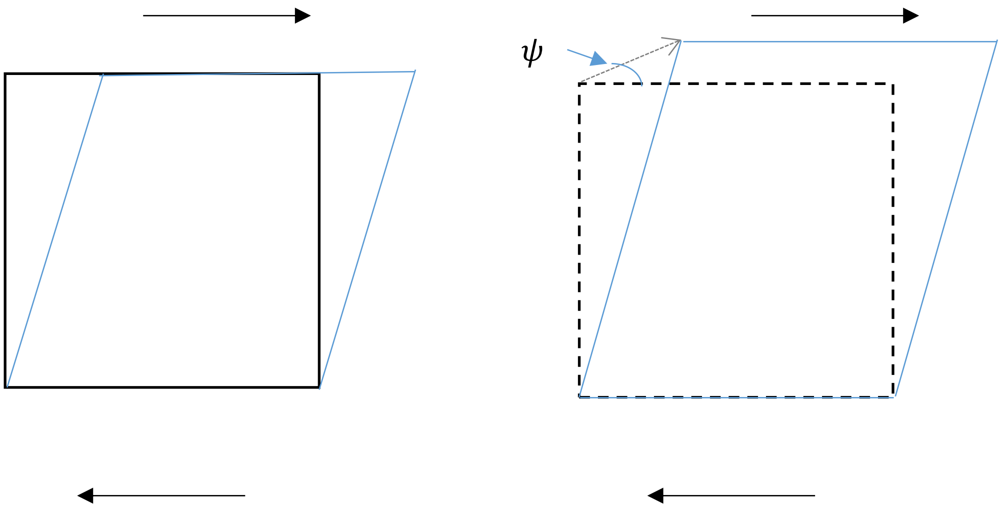


Fig 3.5: Dilatency angle

In the first case (fig 3.5 (a)) the square has undergone distortion and no volumetric strain i.e., no dilation. In the second case fig 3.5 (b)) the square has undergone distortion with volumetric strain, the amount by which this distortion takes place is called dilation angle.

(ψ=38^0 (ref. Jankowiak et al., (2005).)

Table 3.2: CPDM parameters used in this study

K_C	Dilatency angle (ψ)	σ_{bo}/σ_{co}	ϵ	μ
0.67	38^0	1.16	0.1	0.0001

3.5.2 Uniaxial Compression behaviour of concrete:

The stress-strain compression concrete model used in this study was developed by Hsu-Hsu (1994). Fig 3.6 shows the compression stress strain curve developed by Hsu Hsu (1994). This stress strain behaviour is modelled in three stages. The first two stages defines the ascending branch of the curve and the third one descending portion of the curve.

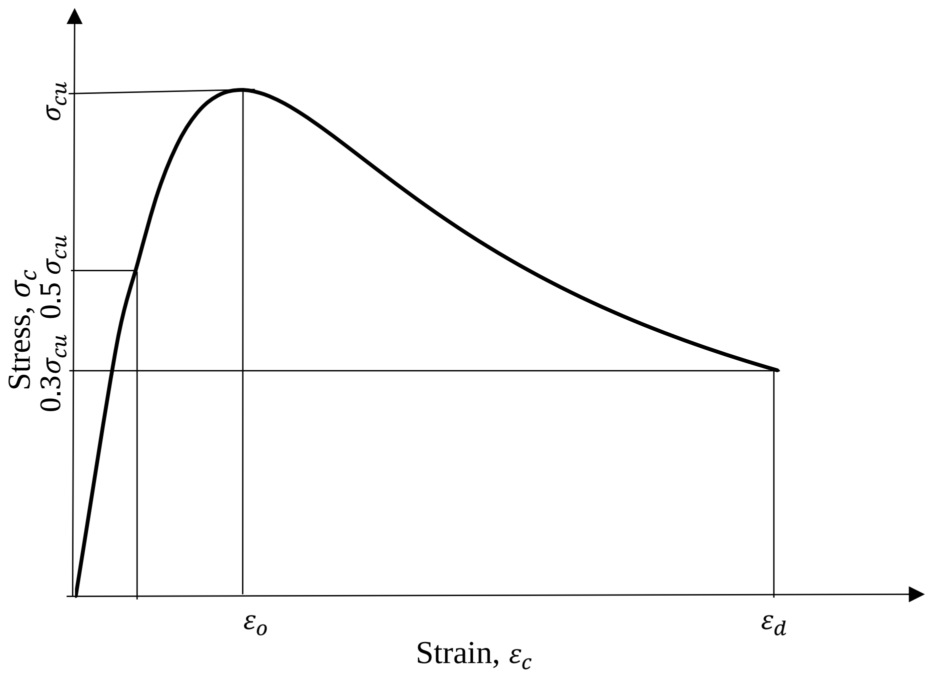


Fig 3.6: Uniaxial Compressive Stress Strain Curve (Hsu Hsu model)

1) In the ascending portion of the branch upto 50% of the ultimate compressive strength (σ_{cu}) , a linear stress strain relationship which obeys Hooke's law is assumed.

$$\sigma_c = E_o \varepsilon_c$$

2) From $0.5\sigma_{cu}$ to the ultimate stress σ_{cu} a non-linear nature of concrete is modelled using the following equations.
3) In the descending portion upto $0.3\sigma_{cu}$ the stress strain curve is modelled.

The Hsu Hsu model (1994) is used to calculate the compression stress strain values only after the yield i.e., 0.5 σ_{cu} in the ascending branch to the $0.3\sigma_{cu}$ in the descending branch.

$$\sigma_c = [\frac{\beta({}^{\varepsilon_c}/_{\varepsilon_o})}{\beta - 1 + ({}^{\varepsilon_c}/_{\varepsilon_o})^{\beta}}]\sigma_{cu}$$

The parameter β defines the shape of the stress strain curve after yielding of concrete at $0.5\sigma_{cu}$

$$\beta = \frac{1}{1 - [{}^{\sigma_{cu}}/_{(\varepsilon_0 E_o)}]}$$

The modulus of elasticity is given by

$$E_o = 1.2431x10^2 + 3.28312x10^3$$

Strain ε_o at the ultimate compressive stress σ_{cu} is given by

$$\varepsilon_o = 8.9x10^{-5}\sigma_{cu} + 2.114x10^{-3}$$

The units in the above equations are in kip/in^2

The inelastic strains are to be entered in the CDPM corresponding to the stresses and it is given by

$$\tilde{\varepsilon}_c^{in} = \varepsilon_c - \varepsilon_{oc}^{el}$$

Where, $\varepsilon_{oc}^{el} = \frac{\sigma_c}{E_c}$ is the elastic strain corresponding to the un-yielded material.

ε_c is the total strain corresponding to the particular stress value.

3.5.3 Uniaxial Tension of Concrete:

The complete tensile stress-strain behaviour of concrete which accounts for strain-softening, tension stiffening is necessary for the simulation in abaqus. Hence we have to input the values of young's modulus E_0, stress σ_t, cracking strain ε_{cr} and the damage parameter values $d_t$corresponding to the grade of concrete used. The cracking strain can be calculated from the total stain using the following equation

$$\tilde{\varepsilon}_t^{ck} = \varepsilon_t - \varepsilon_{ot}^{el}$$

ε_t is the total stain and $\varepsilon_{ot}^{el} = \frac{\sigma_t}{E_0}$ is the elastic stain for the un-yielded material.

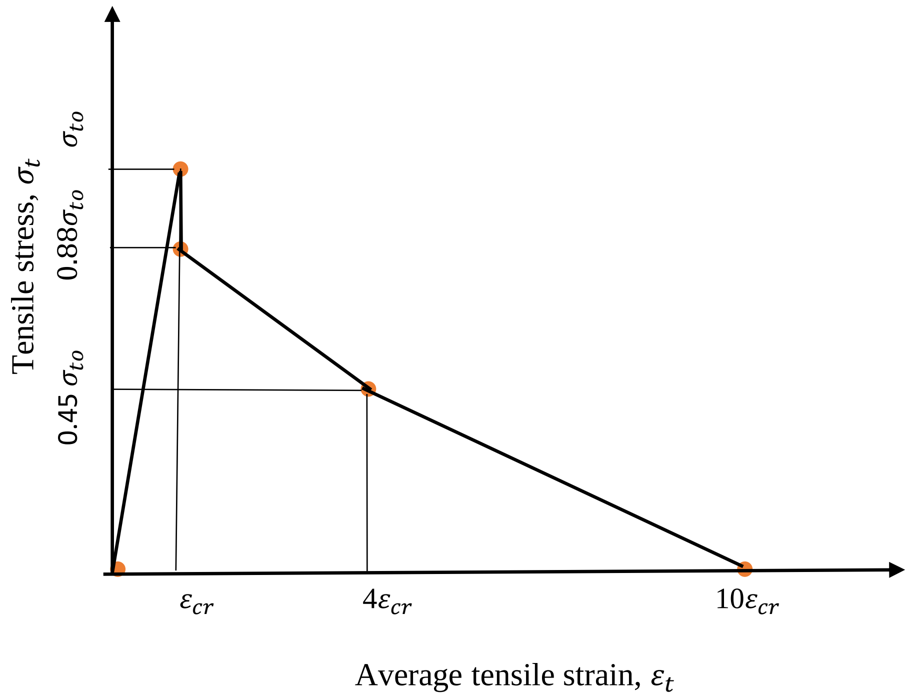


Fig 3.7: Nayal and Rasheed Tension Model (2006)

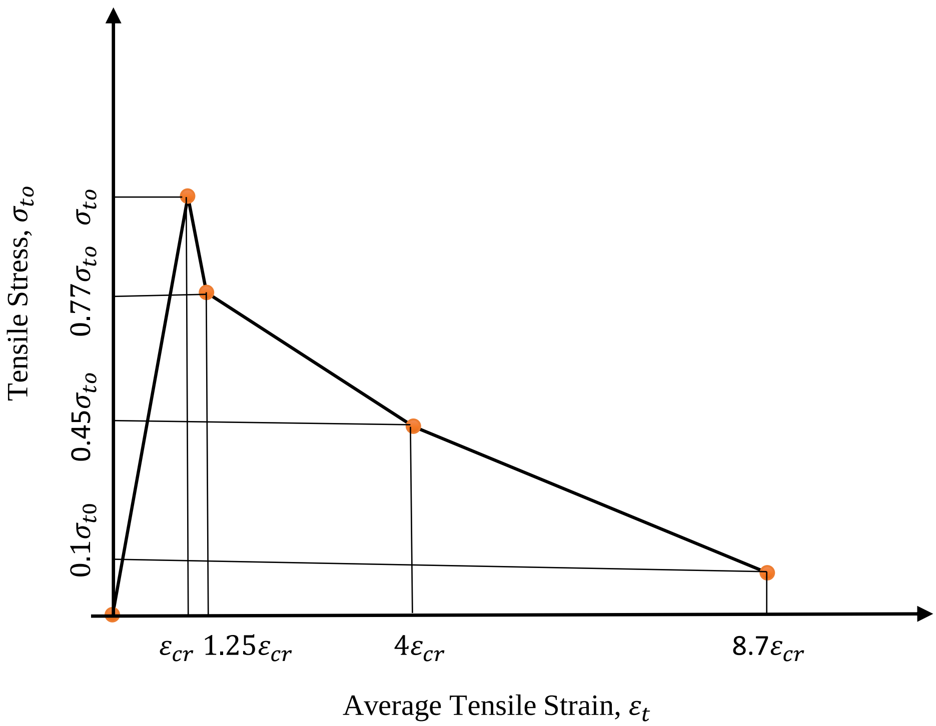


Fig 3.8: Modified Tension Model for Abaqus

The tensile stress strain model of concrete used in this study was developed by Nayal and Rasheed (2006) (fig 3.7). This model resemblance the tension stiffening model required for the CDPM in abaqus. The descending portion of the tensile stress strain is based on the cracking phenomena developed by Gilbert and Warner (1978). The abrupt drop at maximum tensile strain ε_{cr} from ultimate tensile stress σ_{to} to $0.8\sigma_{to}$ as observed by Nayal and Rasheed (2006) is sloped from $(\varepsilon_{cr}, \sigma_{to})$ to $(1.25\varepsilon_{cr},\ 0.77\sigma_{to})$ to eliminate run time errors during analysis in abaqus. Fig 3.6 shows the modified tension stiffening model for abaqus (fig 3.8). Apart from that exactly the same stress strain curve developed by Nayal and Rasheed (2006) is adopted.

3.5.4 Damage Parameter

The plastic strain ε_c^{pl} is proportional to the inelastic strain $\tilde{\varepsilon}_c^{in}$ using a constant b_c(0$< b_c < 1$) and is related to the compressive damage by the following equation model behaves simply as plastic model if these parameters are not defined. The damage variables are considered as non-decreasing values. At any increment during the analysis, the new value of each damage variable is obtained as the maximum between the value at the end of the previous increment and the value corresponding to the current state

$$d_c = 1 - \frac{(\sigma_c/E_c)}{\varepsilon^{pl}(1/b_c - 1) + (\sigma_c/E_c)}$$

Similarly in the tension zone

$$d_t = 1 - \frac{(\sigma_t/E_c)}{\varepsilon^{pl}(1/b_c - 1) + (\sigma_t/E_c)}$$

In the present all the selected piers are modelled using M25 grade concrete using Hsu-Hsu Concrete compression model and Nayal and Rasheed (2006) tension stiffening model. Table 3.3 shows the uniaxial compression and tension stress strain values used to model the piers in the present study.

Table 3.3: Compression and Tension Stress Strain Values of M25 Concrete

Material	M25	Young's Modulus=25×10³MPa	
		Poisons ratio = 0.2	
Concrete Compression		Concrete Compression damage	
Stress (MPa)	Inelastic strain	Damage	Inelastic strain
12.50	0	0	0
16.44	0.00025	0.1024	0.00025
20.00	0.00050	0.1578	0.00050
25.00	0.00150	0.3103	0.00150
23.08	0.00250	0.4482	0.00250
20.00	0.00350	0.5676	0.00350
17.24	0.00450	0.6619	0.00450
15.00	0.00550	0.7333	0.00550
12.82	0.00675	0.7979	0.00675
11.15	0.00800	0.8433	0.00800
Concrete Tension Stiffening		Concrete Tension damage	
Stress (MPa)	Inelastic strain	Damage	Inelastic strain
2.58	0	0	0
2.30	1.21E-05	0.11	1.21E-05
1.99	2.58E-05	0.23	2.58E-05
1.50	0.0002	0.74	0.000193
1.16	0.0003	0.86	0.0003
0.60	0.0006	0.96	0.0006
0.26	0.0008	0.98	0.0008

3.6 SUMMARY

This chapter presents details of selected railway bridge piers and describes about the Concrete Damaged Plasticity model which is used for nonlinear material modelling. It describes about the Abaqus simulation process and different types of elements available in the element library and finally about the modelling of the piers selected for the present study.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTION

The selected railway bridge pier models are analysed using modal analysis and non-linear static (pushover analysis). This chapter presents elastic modal properties of the selected piers, pushover analysis results and discussions. Then a lateral pushover analysis in transverse direction was performed in a displacement control manner.

4.2 RESULTS FROM MODAL ANALYSIS

Modal properties of the selected railway bridge piers were obtained from the linear dynamic modal analysis. Table 4.1 shows the details of the important modes of the bridge in X direction. The table shows that the percentage of mass participation in the first mode is zero and the cumulative mass participating percentage in the first six modes is between 65%-85%. Hence, the higher mode participation in the response of railway bridge pier is significant unlike in regular buildings where only fundamental mode contribution is vital. Table 4.2 shows the cumulative mass participation in X and Y directions for first six modes.

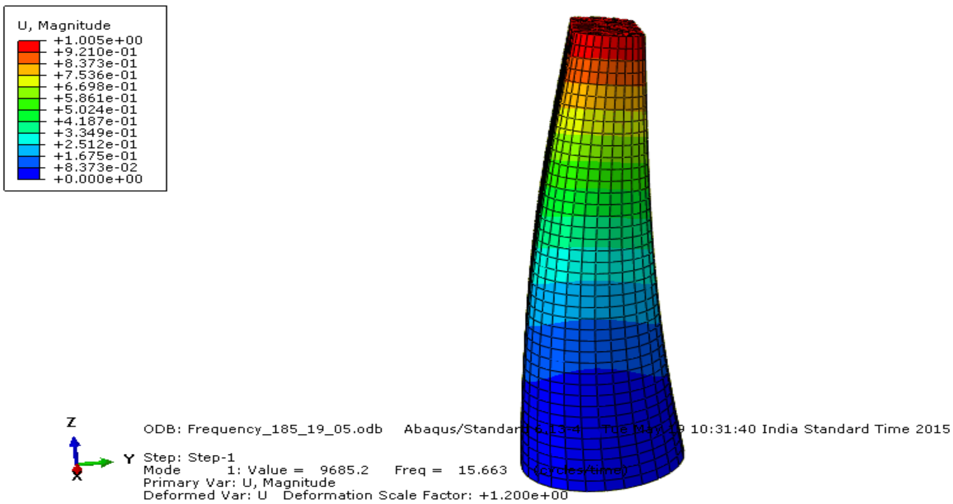
Table 4.1: Elastic Modal Properties for Bridge pier # 6

Mode No.	Frequency (Hz)	Time period (s)	Cumulative Mass Participation (UX)	Cumulative Mass Participation (UY)
1	15.66	0.0638	0	0.44
2	28.77	0.0347	0.48	0.44
3	46.78	0.0214	0.48	0.44
4	56.56	0.0177	0.48	0.67
5	84.80	0.0118	0.48	0.67
6	85.05	0.0118	0.73	0.67

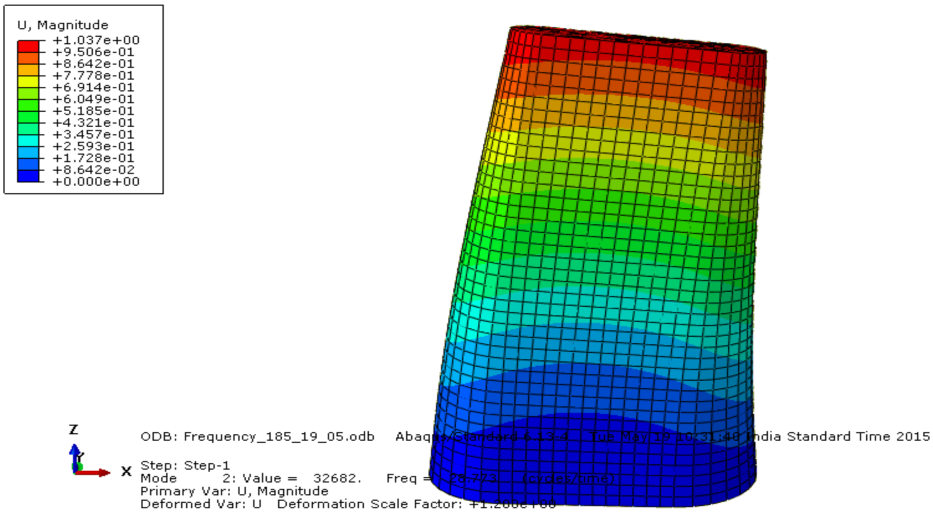
Table 4.2: Cumulative Mass Participation of selected piers in first six mode

Pier No.	UX	UY
1.	0.73	0.66
2.	0.75	0.71
3.	0.61	0.55
4.	0.80	0.75
5.	0.78	0.71
6.	0.79	0.74
7.	0.78	0.75

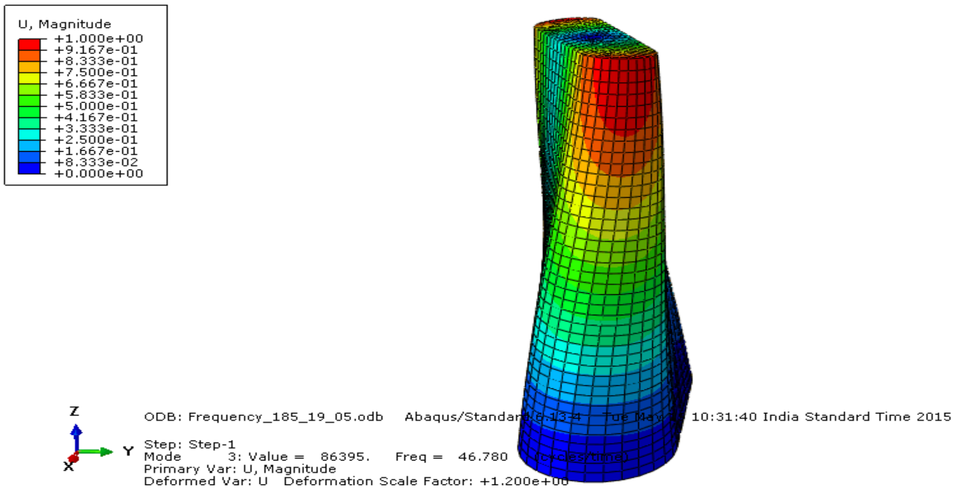
Fig. 4.1 shows the mode shapes of pier# 6 for first six modes. From this figure (also from the Table 4.1) it can be seen that first two fundamental period reflects the translatory motion of the pier in two orthogonal horizontal directions (X- and Y- directions) with significant mass participation although the participating mass ratio in these two modes are below 50% in this case. It can be observed from this figure that torsional mode (Mode# 3) and the rocking mode (Mode# 5) do not contribute anything in the mass participation. Two bending modes (Mode# 4 and Mode# 6) contributes significant amount of mass participation.



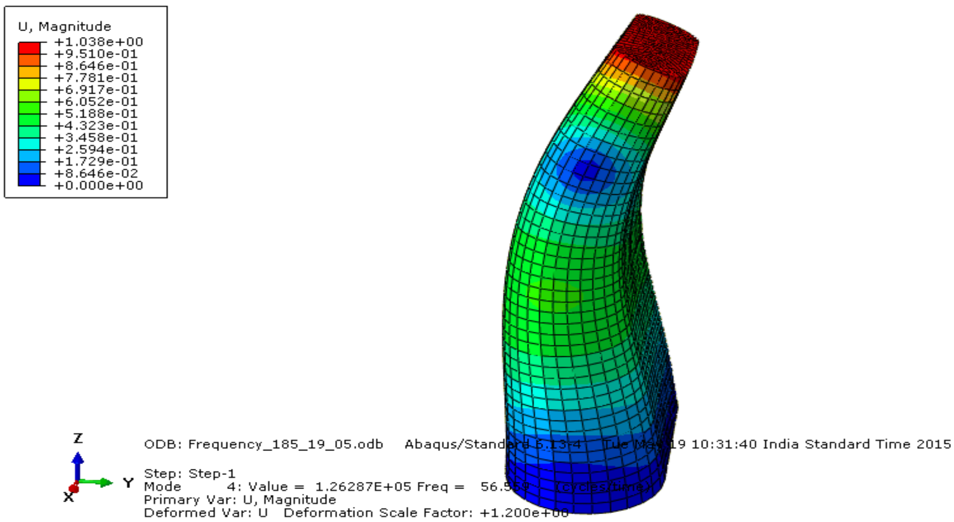
(a) Mode shape # 1



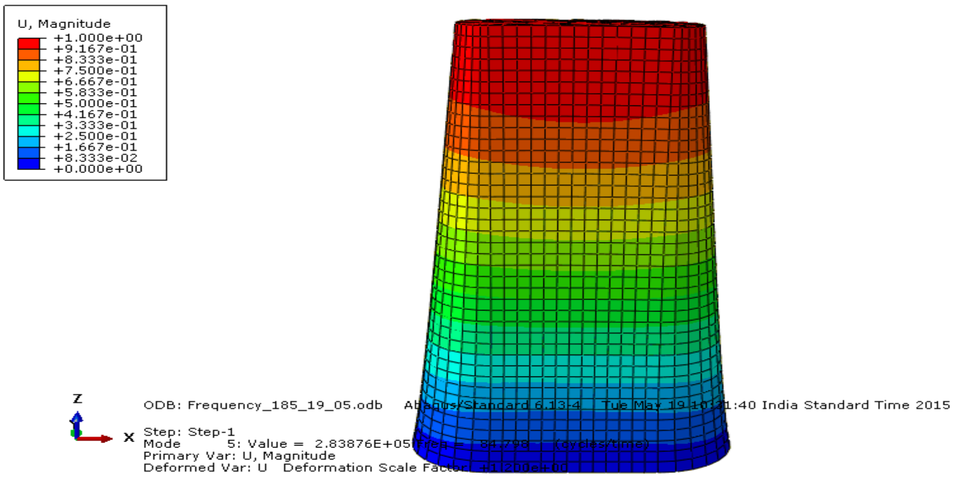
(b) Mode Shape # 2



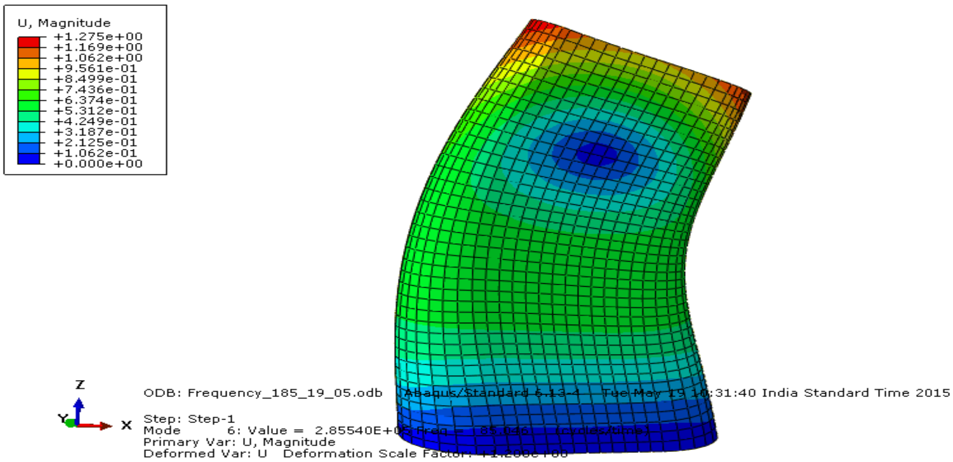
(c) Mode Shape # 3



(d) Mode Shape # 4



(e) Mode Shape # 5



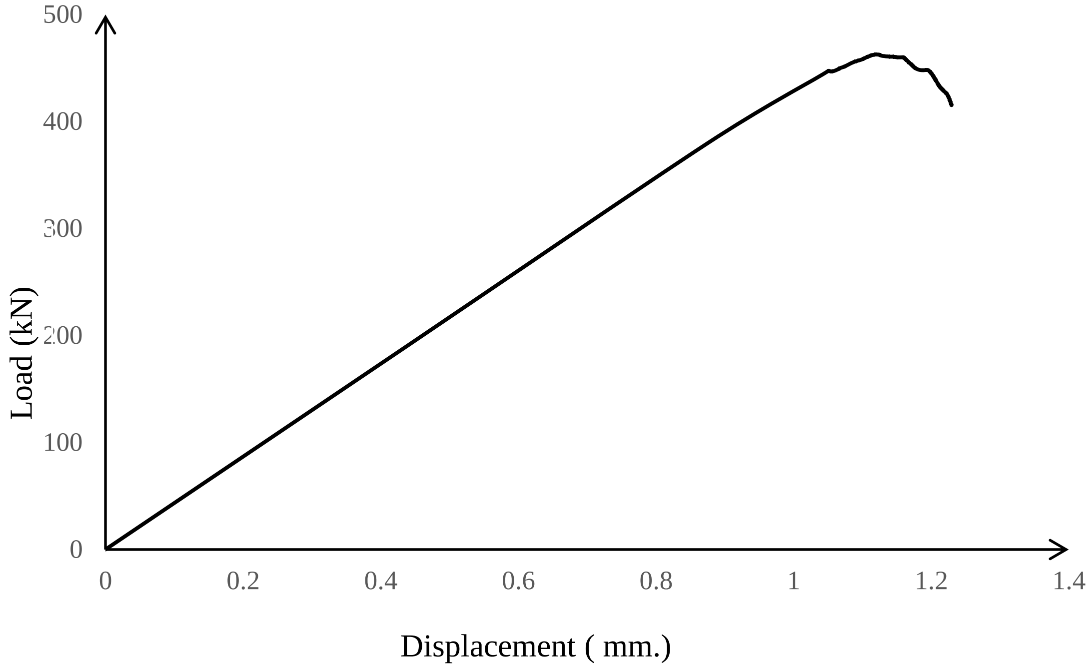
(f) Mode Shape # 6

Fig 4.1: First six Mode shapes of pier # 6

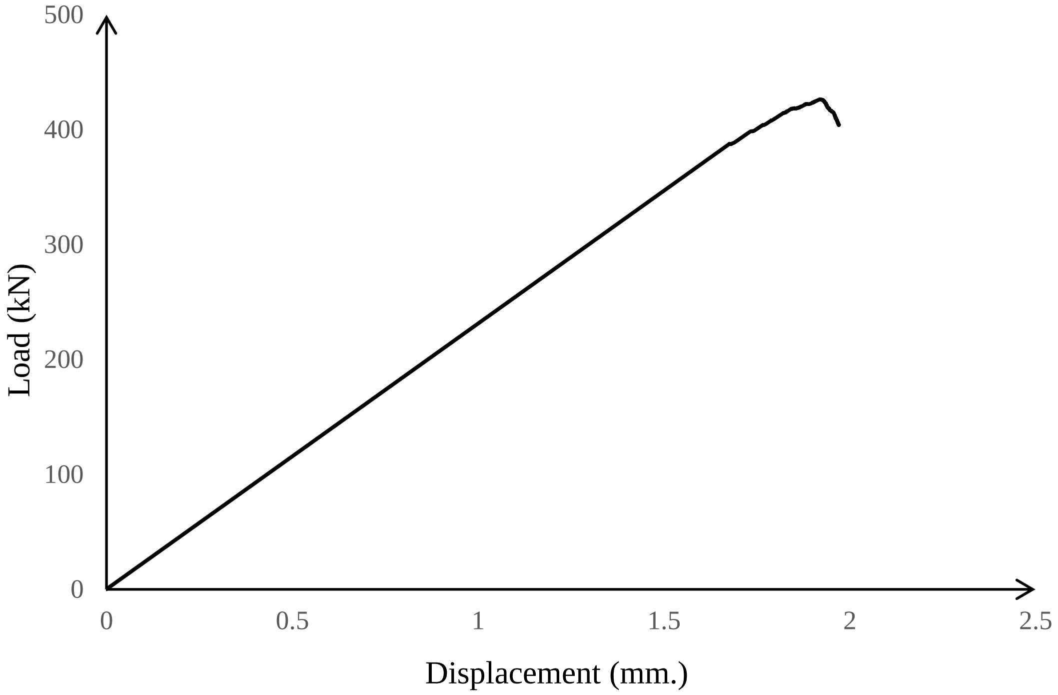
4.3 RESULTS FROM PUSHOVER ANALYSIS

Pushover analysis is carried out on the selected railway bridge piers using displacement controlled method and the capacity curves of the piers are plotted. Fig. 4.2 shows the resulting capacity curve of all the seven bridge piers analysed in this study.

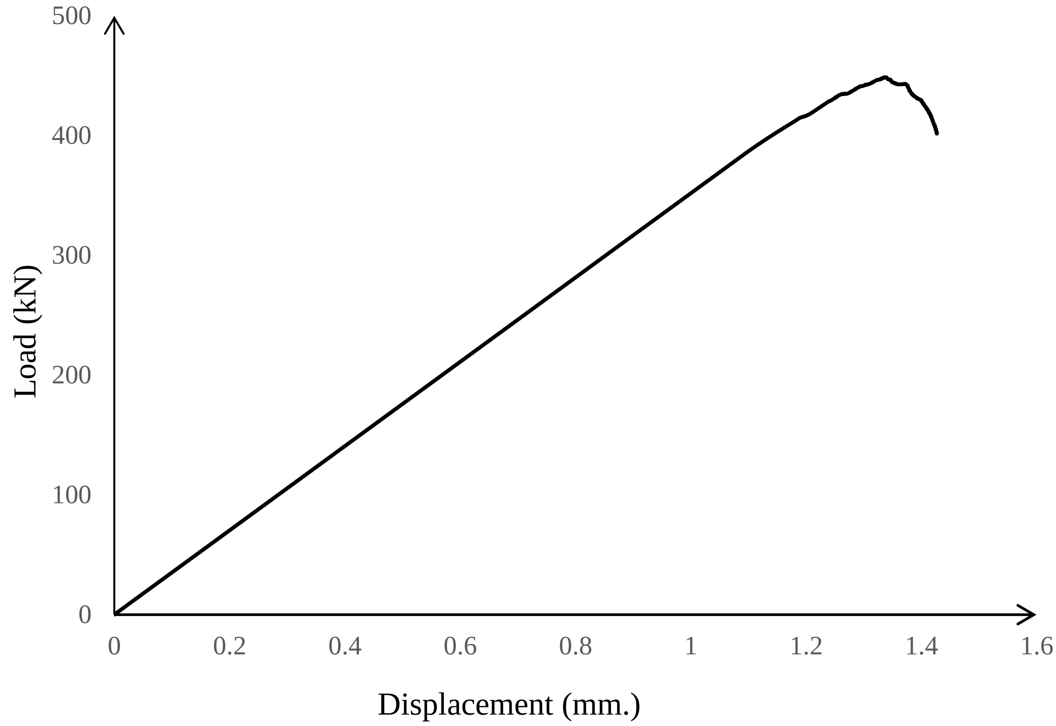
This figures show that the brittle mode of failure of all the bridge piers at ultimate load. This is due to poor energy dissipation capacity of the mass concrete used for building this structures. Table 4.3 presents the summary of the pushover analysis results obtained in the presents study.



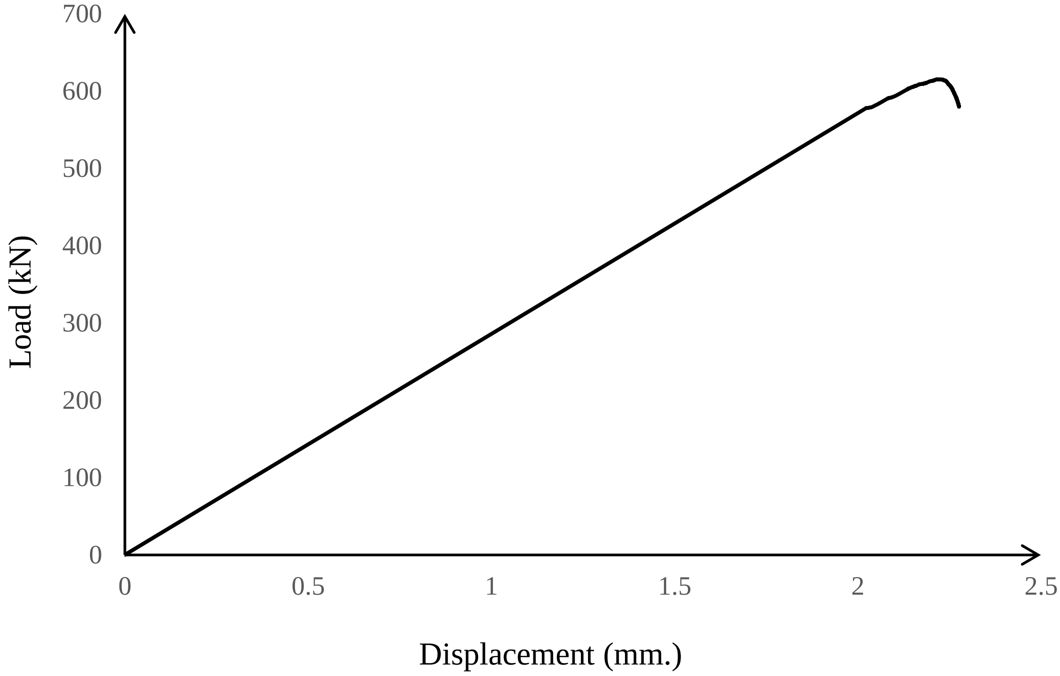
a) Capacity Curve for pier # 1



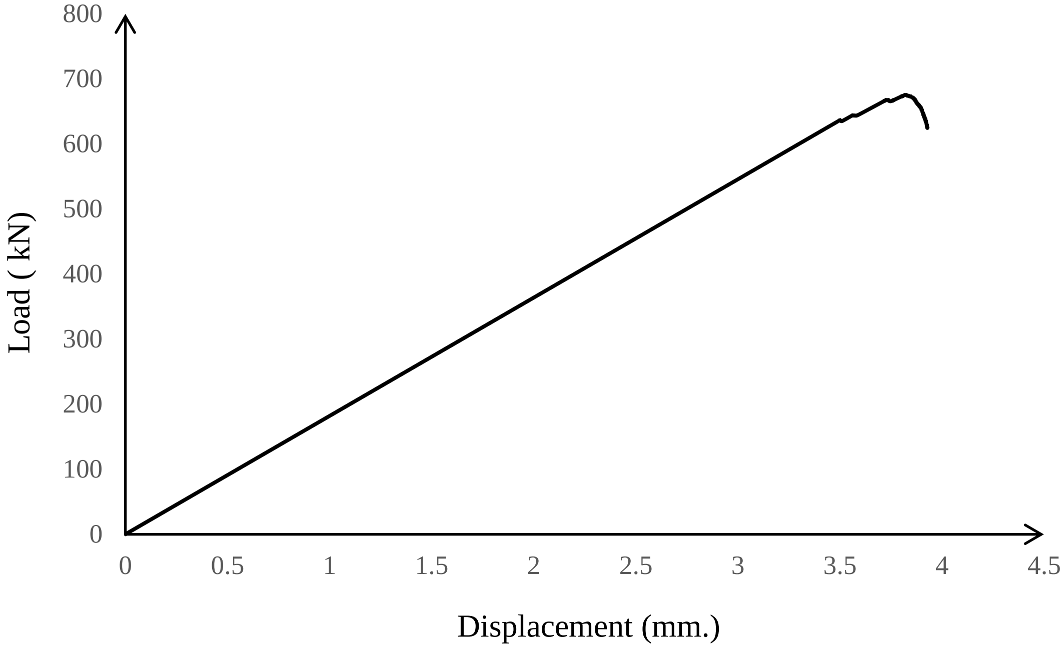
b) Capacity curve of pier # 2



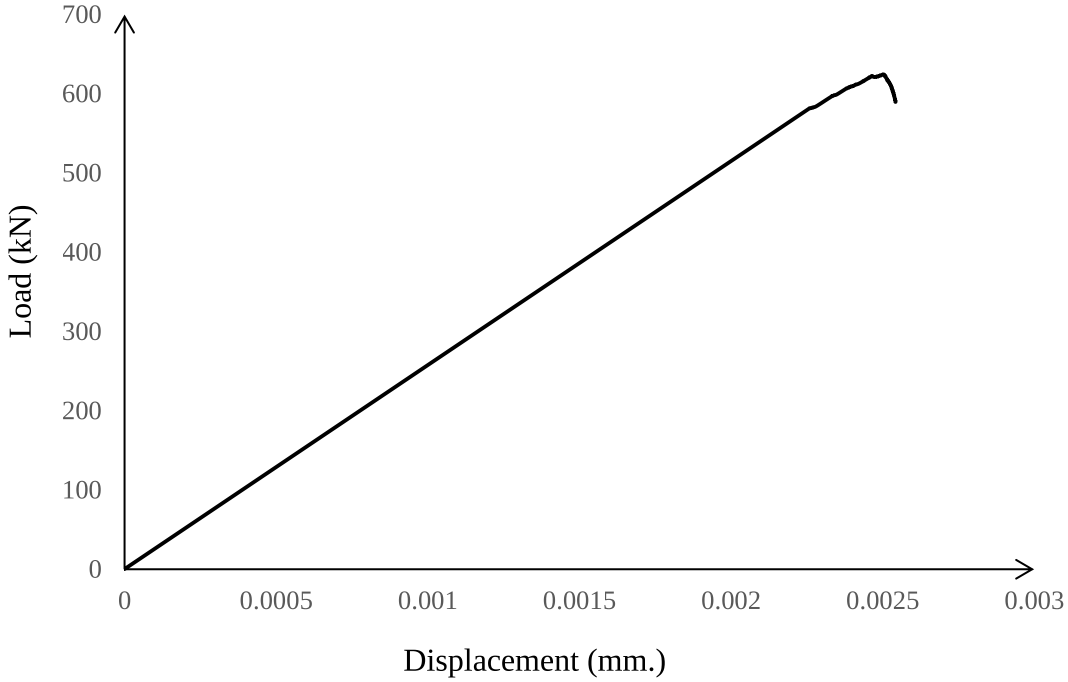
c) Capacity curve of pier # 3



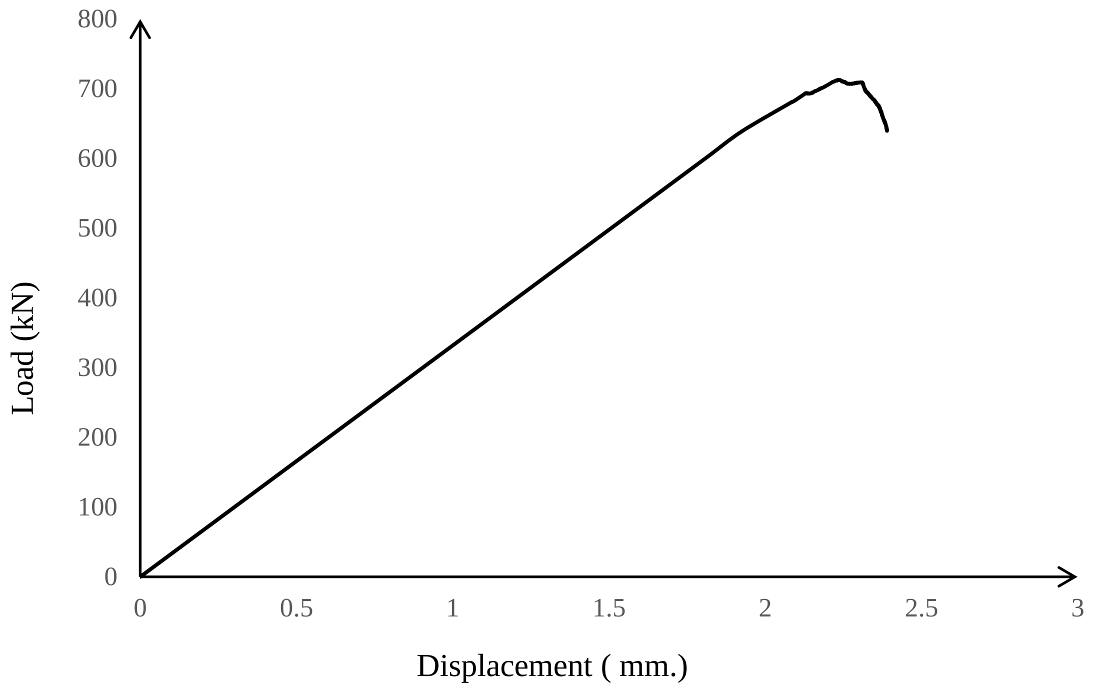
d) Capacity curve for pier # 4



e) Capacity curve of pier # 5



f) Capacity curve of pier # 6



g) Capacity curve for pier # 7

Fig 4.2: Capacity curves of the seven selected piers

Table 4.3: Summary of the pushover analysis results

Pier#	Weight, W (kN)	Base Area (m^2)	Height, h (m)	Top Disp., δ (mm)	Base Shear, V_B (kN)	$\frac{V_B}{W}$	Drift Ratio, $\frac{\delta}{h}$ ($\times 10^{-3}$)
1	2040	14.91	7.500	1.23	462	0.226	0.164
2	3272	18.21	10.500	1.97	426	0.131	0.188
3	2383	15.87	8.402	1.42	448	0.188	0.169
4	4564	26.04	11.250	2.27	615	0.135	0.202
5	10454	36.72	16.875	3.93	675	0.065	0.233
6	5085	27.55	12.000	2.50	624	0.123	0.208
7	6687	30.13	12.375	2.40	712	0.107	0.194

Table 4.3 shows that the displacement limit state of collapse occurs at a base shear range of 6.5-22.6% of the total weight and a drift ratio of 0.0164% to 0.0233%. This table shows that the base shear capacity of the bridge pier (V_B/W) is inversely proportional to the pier height. Fig. 4.3 presents a scatter of (V_B/W) versus pier height.

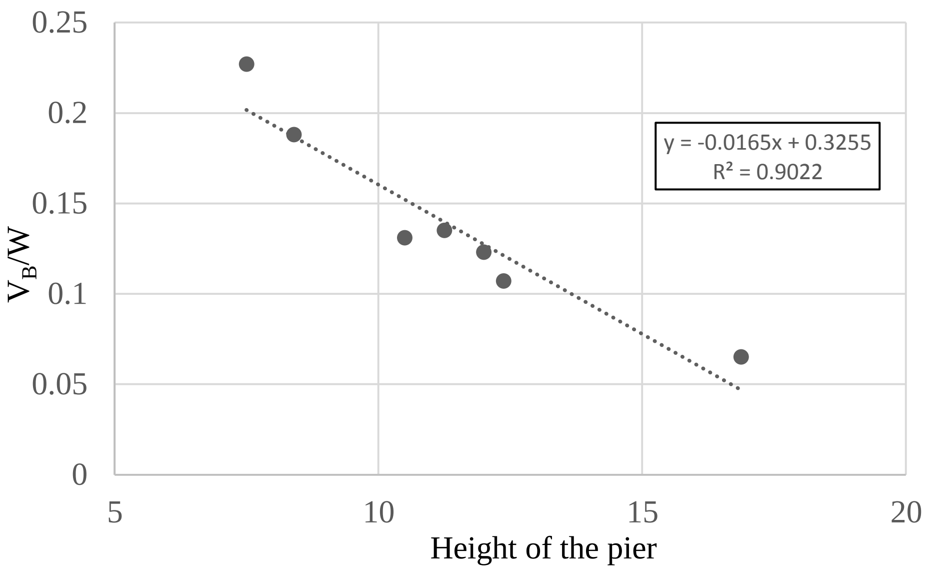


Fig. 4.3: (V_B/W) versus pier height scatter

CHAPTER 5

SUMMARY AND CONCLUSIONS

5.1 SUMMARY

Most of the sub-structures of new railway river bridges in the state of Odisha are built with solid mass concrete gravity piers and abutments. These piers do not have steel reinforcement to bear the load as it does not subject to any tensile stress under regular type of loading. Safety of these piers is of major concern during high magnitude earthquake.

This study aims to assess the vulnerability of the solid gravity bridge piers which forms the important component of railway bridges as the load transfer between substructure and superstructure takes through them. In the present study seven existing piers from the state of Odisha are evaluated using free vibration analysis and nonlinear static (pushover) analysis.

5.2 CONCLUSIONS

The significant conclusion drawn from the present study is as follows:

i) Free-vibration analysis of the bridge pier shows that the first two fundamental modes reflect the translatory motion of the pier in two orthogonal horizontal directions (X- and Y- directions) with mass participation below 50% for both of the two modes.

ii) The participating mass ratio for torsional mode (Mode# 3) and the rocking mode (Mode# 5) found to be zero.

iii) The cumulative mass participation for first six mode is found to be less than 80% for all the selected bridge pier. This indicates the significant contribution of higher modes.

iv) The pushover analysis indicates the brittle mode of failure of all the bridge piers at ultimate load. This is due to poor energy dissipation capacity of the mass concrete used for building these structures.

REFERENCES

1. Abaqus Analysis user manual – Abaqus Version 6.12.
2. Birtel, V., Mark, P., (2006), "Parameterised Finite Element Modelling of RC Beam Shear Failure" *ABAQUS Users' Conference,* 95-104.
3. Chang Su Shim, Chul-Hun Chung, Hyun Ho Kim (2008), "Experimental evaluation of seismic performance of precast segmental bridge piers with a circular solid section", *Engineering Structures,* 30, 3782-3792.
4. Cheng, C. (2007), "Energy dissipation in rocking bridge piers under free vibration tests", *Earthquake Engineering Structural Dynamics,* 36, 503-518.
5. Cho, C., Han, S., Kwon, M. and Lim, C. (2012), "Seismic Performance Evaluation of Reinforced Concrete Columns by Applying Steel Fiber-Reinforced Mortar at Plastic Hinge Region", *Journal of the Korea Concrete Institute*, 24, 241-248.
6. Dassault Systèmes Simulia Corp. Abaqus v. 6.12 [Software].
7. Do Hyung Lee, Eunsoo Choi, Goangseup Zi (2005), "Evaluation of earthquake deformation and performance for RC bridge piers", *Engineering Structures,* 27, 1451-1464.
8. Hsu, L.S., & Hsu, C.-T.T. (1994), "Complete stress-strain behaviour of high-strength concrete under compression", *Magazine of Concrete Research,* 46, 301-312.
9. Huili Wang, Wang, H., Qin, S.F., Xu, W.J., (2011), "Seismic analysis of Prestressed Bridge Pier Based on Fiber Section", *International Conference on Green Buildings and Sustainable Cities,* 21, 354–362.
10. Kim, T.-H., Kim, B.S., Chung, Y.-S., Shin, H.M., (2006), "Seismic performance assessment of reinforced concrete bridge piers with lap splices" *Engineering Structures,* Vol. 28, pp. 935–945.

11. Lee, J., Fenves, GL., (1998), "Plastic-Damage Model for Cyclic Loading of Concrete Structures", *Engineering Mechanics*, 124, 892–900.

12. Lubliner, J., Oliver, J., Oller, S., Oñate, E., (1989), "A plastic-damage model for concrete, *International Journal of Solids and Structures*", 25, 299–326.

13. Memari, A.M., Harris, H.G., Hamid, A.A., and Scanlon, A., (2011), "Seismic Evaluation of Reinforced Concrete Piers in Low to Moderate Seismic Regions", *Electronic Journal of Structural Engineering*, 11, 57-68.

14. Nayal, R., Rasheed, H.A. (2006), "Tension Stiffening Model for Concrete Beams Reinforced with Steel and FRP Bars", *Journal of Materials in Civil Engineering,* 18, 831-841.

15. Rui Faria, Nelson Vila Pouca and Raimundo Delgado (2000) "Seismic behaviour of r/c bridge piers: numerical simulation and experimental validation", *The 12th World Conference on Earthquake Engineering,* No.0673

16. Spyrak, C. C. (1992), "Seismic behavior of bridge piers including soil-structure interaction" *Computers & Structures,* 43, 373-384.

17. Wahalathantri, B.L., Thambiratnam, D.P., Chan, T.H.T., & Fawzia, S. (2011), "A material model for flexural crack simulation in reinforced concrete elements using abaqus", *eddBE2011 proceedings,* 260-264

18. Wang, D.S., Ai, Q.H., Li, H.N., Si, J., and Sun, Z.G. (2008), "Displacement based seismic design of RC bridge piers: Method and experimental evaluation", *The 14th World Conference on Earthquake Engineering* Beijing, China.